Advances in Applied Mathematics and Mechanics
Adv. Appl. Math. Mech., Vol. **7**, No. 2, pp. 180-195

DOI: 10.4208/aamm.2014.m605
April 2015

A High-Order Numerical Method to Study Three-Dimensional Hydrodynamics in a Natural River

Luyu Shen[1,2], Changgen Lu[1,*], Weiguo Wu[3] and Shifeng Xue[3]

[1] *College of Marine Sciences, Nanjing University of Information Science and Technology, Nanjing 210044, China*
[2] *College of Atmospheric Sciences, Nanjing University of Information Science and Technology, Nanjing 210044, China*
[3] *Department of Engineering Mechanics, China University of Petroleum, Dongying 266555, China*

Received 1 May 2014; Accepted (in revised version) 20 August 2014

Abstract. A high-order numerical method for three-dimensional hydrodynamics is presented. The present method applies high-order compact schemes in space and a Runge-Kutta scheme in time to solve the Reynolds-averaged Navier-Stokes equations with the k-ϵ turbulence model in an orthogonal curvilinear coordinate system. In addition, a two-dimensional equation is derived from the depth-averaged momentum equations to predict the water level. The proposed method is first validated by its application to simulate flow in a 180° curved laboratory flume. It is found that the simulated results agree with measurements and are better than those from SIMPLEC algorithm. Then the method is applied to study three-dimensional hydrodynamics in a natural river, and the simulated results are in accordance with measurements.

AMS subject classifications: 65M06, 76F60, 35Q35

Key words: Compact scheme, three-dimensional hydrodynamics, k-ϵ turbulence model, orthogonal curvilinear coordinate system, Runge-Kutta scheme.

1 Introduction

Numerical simulation of fluid motions in natural rivers, reservoirs, bays or harbors is a challenging issue in computational fluid dynamics (CFD), because of their turbulent nature, multi-scale secondary currents, irregular geometry, etc. To essentially solve this problem, one has to use the three-dimensional hydrodynamic models, since

*Corresponding author.
Email: shenluyu99@foxmail.com (L. Shen), cglu@nuist.edu.cn (C. Lu), mailtowwg@gmail.com (W. Wu), xuesf@126.com (S. Xue)

the one-dimensional and two-dimensional models cannot properly describe the three-dimensional flow features such as secondary currents and three-dimensional turbulence [1]. The use of suitable mathematic models is one of the key issues in the three-dimensional hydrodynamic study. The Reynolds-averaged Navier-Stokes (RANS) equations with the k-ϵ turbulence model have been shown to describe three-dimensional flow adequately in many hydraulic applications. Demuren and Rodi [2] used a modified k-ϵ model to study flow in a meandering channel. Ye and McCorquodale [3] simulated three-dimensional free surface flow with a k-ϵ model taking account into anisotropic effects. Wu et al. [4] simulated flow and sediment transport in a 180° bend with the k-ϵ model. More recent simulations are presented by Rüther and Olsen [5], Zhang and Shen [6] and Khosronejad et al. [7].

In the previous studies, the normally used numerical methods of three-dimensional hydrodynamic models are SIMPLE algorithm and its variants such as SIMPLEC, SIMPLEX, and PISO. Zhang and Shen [6] used SIMPLEC algorithm to investigate the river transport of pollution. Feurich and Olsen [8] used SIMPLE algorithm for computing the water flow in a non-orthogonal, structured grid with a non-staggered variable placement. Zeng et al. [9] used a fractional step method to solve the RANS equations in generalized curvilinear coordinates. The advantage of using SIMPLE algorithm is to keep numerical conservation, and the drawback is difficult to extend the method with high-order accuracy which may not match the growing need of hydrodynamics simulations. To remedy the drawback, we will use high-order compact schemes to do spatial discretization. Compact schemes, which were proposed from a generalization of the classical Padé schemes by Lele [10], are high-order schemes extensively applied in Direct Numerical Simulation (DNS) [11,12] and Large Eddy Simulation (LES) [13,14] for turbulence simulation. In our research, the compact schemes are extended to study three-dimensional hydrodynamics.

In our work, a high-order upwind-biased compact scheme [15] is used for convection term discretization on purpose to suppress the spurious waves. For temporal discretization, a Runge-Kutta time stepping scheme is employed. To apply the uniform mesh-based compact schemes for problems with complex geometry, a coordinate transformation is introduced to transform the governing equations in the physical space into those in the curvilinear coordinate system so that the computational domain is always regular and that a uniform mesh can be used. To predict the water level, a two-dimensional equation is derived from the depth-averaged momentum equation following the work of Wu et al. [4]. The flow in a 180° curved laboratory flume [16] is simulated to evaluate the performance of the new method. Then, the proposed method is applied to study the three-dimensional hydrodynamics in a natural river.

2 Governing equations and boundary conditions for flow field

The three-dimensional hydrodynamic model is composed of the incompressible Reynolds-averaged Navier-Stokes equations and standard k-ϵ turbulence model. The continuity and momentum equations in an orthogonal curvilinear coordinate are written